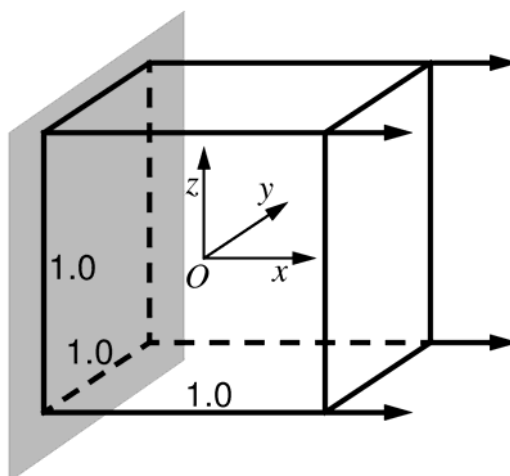


# Description of the elasto-plastic material routine SGDI

Folco Casadei  
Vegard Aune  
Georgios Valsamos  
Martin Larcher

2015

Report EUR 27434 EN



European Commission  
Joint Research Centre  
Institute for the Protection and Security of the Citizen

Contact information

Georgios Valsamos

Address: Joint Research Centre, Via Enrico Fermi 2749, TP 480, 21027 Ispra (VA), Italy

E-mail: [georgios.valsamos@jrc.ec.europa.eu](mailto:georgios.valsamos@jrc.ec.europa.eu)

Tel.: +39 0332 78 9004

Fax: +39 0332 78 9049

<http://ipsc.jrc.ec.europa.eu/>

<http://www.jrc.ec.europa.eu/>

This publication is a Technical Report by the Joint Research Centre of the European Commission.

**Legal Notice**

This publication is a Technical Report by the Joint Research Centre, the European Commission's in-house science service. It aims to provide evidence-based scientific support to the European policy-making process. The scientific output expressed does not imply a policy position of the European Commission. Neither the European Commission nor any person acting on behalf of the Commission is responsible for the use which might be made of this publication.

JRC97557

EUR 27434 EN

ISBN 978-92-79-51226-1 (PDF)

ISSN 1831-9424 (online)

doi:10.2788/403240

Luxembourg: Publications Office of the European Union, 2015

© European Union, 2015

Reproduction is authorised provided the source is acknowledged.

Printed in Italy

# Description of the elasto-plastic material routine SGDI

F. Casadei, Vegard Aune, Georgios Valsamos, Martin Larcher

September 19, 2015

These notes are based upon the state of the SGDI material routine at the time of this writing (September 2015). The text of the routine has been slightly edited (but without substantial changes) in order to simplify the description.

## 1 Introduction

We describe the FORTRAN subroutine SGDI for rate-independent deviatoric plasticity with von Mises yield criterion and isotropic hardening, which corresponds to the VM23 material type in the EUROPLEXUS code (abbreviated as EPX in the following). EPX [1] is a Finite Element code for the numerical simulation of fast transient dynamic events in fluid-structure system jointly developed by the French Commissariat à l’Energie Atomique et aux Energies Alternatives (CEA Saclay) and by the Joint Research Centre of the European Commission (JRC Ispra).

The subroutine is part of the FORTRAN90 module M\_MATERIAL\_VM23. In order to simplify the description of the routine and of the material model, the parts dealing with Gauss point failure and element erosion have been neglected.

## 2 The material routine

### 2.1 Generalities

The SGDI material routine is quite general and is used in conjunction with a variety of Finite Element (FE) types in EPX, including both 2D and 3D cases and dealing with either continuum or structural (plate, shell, beam or bar) formulations.

- For 2D continua, the routine covers the classical plane strain, axisymmetric and plane stress cases. 2D continuum elements lie in the  $xy$  plane and stresses are expressed in the global (fixed) reference frame. The out-of-plane stress is denoted  $\sigma_z$  except in the axisymmetric case where it is usually denoted  $\sigma_\theta$ . In 2D axisymmetric calculations the mesh must be constructed in such a way that the axis of symmetry coincides with the  $y$ -axis (i.e. with the “vertical” global axis). The code assumes one radians (not  $2\pi$  radians) extension of the model in the azimuthal direction. Therefore, the resulting total masses, forces etc. printed by the code will be  $1/2\pi$  times the real (physical) values.
- 2D beams/shells are assumed to lie in the  $xz$  plane, so that for them the out-of-plane stress is  $\sigma_y$ . For these elements the stresses are expressed in a local convected reference frame ( $x$  is taken along the length of the element), which rotates rigidly in the  $xz$ -plane together with the element. The stress through the element thickness ( $\sigma_z$  by convention) is considered to be zero or very small compared with the other stress components.

- If, in addition to this, a zero stress condition normal to the plane ( $y$  direction) is also imposed, then the element no longer represents a shell, but a beam. In the Euler-Bernoulli beam theory the shear stresses are not considered while they are present in Timoshenko beams. (At the moment, no Timoshenko beam element and no beam element subjected to torsion for use with the VM23 material is implemented in EPX.)
- Bars and cables are similar to beams but do not present bending. Bars only resist in traction and compression, while cables only resist in traction. In both cases, the only non-zero stress component is the longitudinal stress ( $\sigma_x$ ), and this both in 2D and in 3D.
- The case of 3D continuum elements is the most general one. All six stress components are present and must be treated. Like for 2D continuum elements, the stresses are expressed in the global (fixed) reference frame.
- For 3D plates/shells, the stress through the element thickness ( $\sigma_z$  by convention) is assumed to be negligible. For these elements the stresses are expressed in a local convected reference frame ( $xy$  lies in the plane of the element) which rotates rigidly in 3D space together with the element. The transverse shear stresses  $\tau_{yz}$  and  $\tau_{xz}$  may or may not be present, depending on the particular shell theory assumed (Mindlin or Kirchhoff theory, respectively).

In order to treat all these different cases the routine makes use of two integer values, **ITAU** and **IPLANC**, which are passed through the exchange list. **ITAU** is the number of shear components to be treated, which can be 0, 1 or 3. **IPLANC** is the number of ‘plane’ (or zero/negligible) stress conditions and can be 0, 1 or 2. The various possible combinations are summarized in Table 1. Note that in the particular case of 3D shells developed at CEA (e.g. Q4GS) **IPLANC** may assume the value 3. This is equivalent to the case of **IPLANC** equal 1 (zero stress through the shell thickness) but is used as a flag to activate a special treatment of the zero-stress condition (see details below).

ITAU	IPLANC	Case	Components treated
0	1	2D Euler-Bernoulli plane strain beam or axisymmetric shell	$\sigma_x, (\sigma_y \text{ or } \sigma_\theta), \tau_{xy} = 0, \sigma_z \approx 0$
0	2	2D Euler-Bernoulli uniaxial stress beam or 2D/3D bar	$\sigma_x, \sigma_y \approx 0, \tau_{xy} = 0, \sigma_z \approx 0$
1	0	2D Plane strain or axisymmetric	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
1	1	2D Plane stress or 3D Kirchhoff plate/shell	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
1	3	3D Kirchhoff plate/shell (CEA approach: $\sigma_z = 0$ )	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z = 0$
3	0	3D Continuum	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, \tau_{yz}, \tau_{xz}$
3	1	3D Mindlin plate/shell	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0, \tau_{yz}, \tau_{xz}$
3	3	3D Mindlin plate/shell (CEA approach: $\sigma_z = 0$ )	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z = 0, \tau_{yz}, \tau_{xz}$

Table 1: Valid combinations of **ITAU** and **IPLANC**

The element types which can use the VM23 material and the associated values of **ITAU** and **IPLANC** are summarized in Table 2 for 2D elements and in Table 3 for 3D elements.

## 2.2 The exchange list

In EPX a material constitutive routine can be seen as a black box which basically receives in input the (old) stresses, the (old) hardening variables and the strain increments. The routine computes and returns the new stresses and it updates the hardening variables. Note that in the plane stress or in the uniaxial stress cases, the routine should also (re-)compute the lateral strain increments ( $\Delta\epsilon_z$  or  $\Delta\epsilon_z$  and  $\Delta\epsilon_y$ , respectively) resulting from the zero-stress condition(s) imposed.

The ‘old’ stresses are those at time  $t^n$ , while the ‘new’ ones are those at time  $t^{n+1} = t^n + \Delta t$ ,  $\Delta t$  being the time increment, which is usually very small in explicit codes such as EPX.

Element	ITAU	IPLANC	Description
FUN2	0	2	2-node bar
ED01	0	1	2-node Euler beam (plane strain) or conical shell (axisymmetric)
	0	2	2-node Euler beam (plane stress means uniaxial stress here)
ED41	0	1	4-node Euler beam (plane strain) or conical shell (axisymmetric)
	0	2	4-node Euler beam (plane stress means uniaxial stress here)
Q41	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q42	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q41N	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q42N	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q41L	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q42L	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q42G	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
Q92	1	0	9-node quadrilateral, plane strain or axisymmetric
	1	1	9-node quadrilateral, plane stress
Q93	1	0	9-node quadrilateral, plane strain or axisymmetric
	1	1	9-node quadrilateral, plane stress
Q92A	1	0	9-node quadrilateral, axisymmetric
Q95	1	0	9-node quadrilateral, plane strain or axisymmetric
	1	1	9-node quadrilateral, plane stress
CAR1	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
CAR4	1	0	4-node quadrilateral, plane strain or axisymmetric
	1	1	4-node quadrilateral, plane stress
TRIA	1	0	3-node triangle, plane strain or axisymmetric
	1	1	3-node triangle, plane stress

Table 2: ITAU and IPLANC for 2D elements which can use the VM23 material

The routine is called for each Gauss point (GP) of an element that uses the VM23 material. Within the routine itself, we are not concerned about the nature of the element calling it, since ITAU and IPLANC are sufficient to precisely determine which components of stress, strain etc. should be treated.

The exchange list of the routine reads

```

SUBROUTINE SGDI (ITAU, IPLANC, XMAT, DEPS, SIG, EPST, ECR, PI, CSON,
>                FAIL_FLAG)

```

where ITAU and IPLANC have been already described above, while the meaning of the other variables is as follows:

- XMAT is an array containing the material properties (constants) for this particular instance of the VM23 material:

```

* xmat(1) : Initial density
* xmat(2) : Young's modulus
* xmat(3) : Poisson's coefficient

```

Element	ITAU	IPLANC	Description
FUN3	0	2	2-node bar
COQI	1	1	3-node triangular Kirchhoff plate/shell
DKT3	1	3	3-node triangular Kirchhoff shell
C272	3	0	27-node hexahedron
C273	3	0	27-node hexahedron
C81L	3	0	8-node hexahedron
C82L	3	0	8-node hexahedron
CUB6	3	0	8-node hexahedron
CUB8	3	0	8-node hexahedron
CUBE	3	0	8-node hexahedron
PR6	3	0	6-node prism
PRIS	3	0	6-node prism
TETR	3	0	4-node tetrahedron
CQD4	3	1	4-node quadrilateral Mindlin shell
CQD9	3	1	9-node quadrilateral Mindlin shell
CQD3	3	1	3-node triangular Mindlin shell
CQD6	3	1	6-node triangular Mindlin shell
T3MC	3	1	3-node triangular Mindlin shell
DST3	3	3	3-node triangular Mindlin shell
Q4GS	3	3	4-node quadrilateral Mindlin shell
T3GS	3	3	3-node triangular Mindlin shell
Q4GR	3	3	4-node quadrilateral Mindlin shell
QPPS	3	3	4-node quadrilateral Mindlin shell

Table 3: ITAU and IPLANC for 3D elements which can use the VM23 material

```

* xmat(4) : Elastic limit (initial yield stress)
* xmat(5) : Chosen failure criterion
*          (0=NONE,1=VMIS,2=PEPS,3=PRES,4=PEPR ...)
* xmat(6) : Chosen failure limit value
* xmat(7) : N. of stress/slope points on the traction curve
* xmat(7+2*i-1) : i-th stress (i=1,...,xmat(7))
* xmat(7+2*i ) : i-th slope (dsig/deps)

```

- DEPS are the (natural) strain increments, which are typically computed in the (calling) element routine. However, in the plane stress or uniaxial stress cases the lateral components ( $\Delta\epsilon_z$  or  $\Delta\epsilon_z$  and  $\Delta\epsilon_y$ , respectively) must be computed by the constitutive routine since they result from the zero stress condition(s) imposed. Note that, as concerns the shear components, the “engineering” or gamma values (i.e. twice the epsilon values) are passed:

```

* deps(1) --> deps1 : x-strain increment
* deps(2) --> deps2 : y-strain increment
* deps(3) --> dgam12 : xy-strain increment (gamma=2*epsilon)
* deps(4) --> deps3 : z-strain increment
* deps(5) --> dgam23 : yz-strain increment (gamma=2*epsilon)
* deps(6) --> dgam13 : xz-strain increment (gamma=2*epsilon)

```

- SIG are the old stresses. The organization of components is the same as for DEPS:

```

* sig(1) --> sig1 : x-stress
* sig(2) --> sig2 : y-stress

```

```

* sig(3)  --> tau12  : xy-stress
* sig(4)  --> sig3   : z-stress
* sig(5)  --> tau23  : yz-stress
* sig(6)  --> tau13  : xz-stress

```

- EPST are the total natural strains and use the same organization as SIG and as DEPS (the shear components are the engineering or gamma values, i.e. twice the epsilon values). These are already the values at the end of the time step, except for the cases of plane stress or uniaxial stress (IPLANC greater than 0), in which they are re-computed inside the routine. Note that, since an incremental formulation of the constitutive law is assumed, the total strains do not intervene in the solution, but their values can be useful in the interpretation of results.

- ECR are the (generalized) hardening parameters:

```

* ecr(1) : Hydrostatic pressure (1/3(sx+sy+st))
* ecr(2) : Equivalent stress (von Mises)
* ecr(3) : Equivalent plastic strain
* ecr(4) : Current yield stress
* ecr(5) : Sound speed
* ecr(6) : Failure flag (0=virgin GP, 1 = failed GP)

```

- PI are the new stresses, to be returned by the routine, and use the same organization as SIG.
- CSON is the sound speed, to be returned by the routine.
- FAIL\_FLAG is a flag indicating whether the Gauss point is virgin or failed.

With the observations already presented above, the `INTENT` declaration for the exchange list variables is:

```

REAL(8), INTENT(IN) :: XMAT(*), SIG(*)
REAL(8), INTENT(OUT) :: PI(*), CSON
REAL(8), INTENT(INOUT) :: EPST(*), DEPS(*), ECR(*)
INTEGER, INTENT(IN) :: ITAU, IPLANC
LOGICAL, INTENT(INOUT) :: FAIL_FLAG

```

Note the FORTRAN77 declaration style: the asterisk indicates an array of unknown dimension (length), i.e. the number of components is not checked by the compiler. It is the programmer's responsibility to make sure that non-existing components of `SIG`, `DEPS` etc. are neither used (read), since the value would be essentially random, nor updated (written), since this could cause a memory fault. For the shear components, this can be done by using the value of `ITAU`.

## 2.3 Initializations

The routine starts by extracting the material constants from the `XMAT` array. The values are placed in some local scalar variables for ease of reading and of subsequent use:

```

YMX = XMAT(2)
SO = XMAT(4)
XNU = XMAT(3)

```

```

FAIL_CRIT = XMAT(5)
FAIL_LIMIT = XMAT(6)
T1 = 1.DO + XNU
T2 = 1.DO - 2.DO*XNU
T3 = 1.DO - XNU
CS = YMX / XMAT(1)
D = XNU*YMX / (T1*T2)
G2 = YMX / T1
G3 = G2*1.5D0
CSO = 0.DO ! in case of failure we have a default value ...

```

Thus:

- YMX is Young's modulus  $E$
- SO is the initial yield stress  $\sigma_{Y0}$
- XNU is Poisson's modulus  $\nu$
- T1 is  $(1 + \nu)$
- T2 is  $(1 - 2\nu)$
- T3 is  $(1 - \nu)$
- CS is  $(E/\rho)$ , to be used next for the calculation of the sound speed
- D is Lamé's first constant  $\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}$
- G2 is twice Lamé's second constant  $2\mu$  (or  $2G$ ), i.e. twice the shear modulus  $G$
- G3 is  $3\mu$  (or  $3G$ )

Next, the routine prepares some local variables which will be used in the following to access the material's traction curve, in particular to compute the (current) tangent slope. The necessary information is stored in the **XMAT** array of the material properties.

The material's uniaxial traction curve ( $\sigma$ - $\epsilon$  diagram) is assumed to be composed by a sequence of (straight) line segments, i.e. a "multi-linear" approximated representation of the experimental curve is adopted. The first segment represents the elastic behaviour, while the following segments represent the elasto-plastic part of the curve. Any number of segments (at least one) can be used. However, the slope of the successive segments must decrease monotonically. The (final) slope can be zero (perfectly plastic behaviour), but it cannot become negative since the routine is unable to treat material softening.

In **XMAT(7)** is contained the number of segments  $N$  composing the traction curve. Then, in the following positions are stored  $2N$  values, each couple representing the end of a segment of the traction curve. In the input file, the user specifies, for each point (end of a segment) of the traction curve, a couple of values  $(\sigma, \epsilon)$ , i.e. first the (true) stress and then the corresponding (total natural) strain. The first couple of values are the initial yield stress  $\sigma_{Y0}$  (which must also be provided separately as a material property **ELAS** in the input file, with the same value), and the corresponding strain  $\epsilon_Y = \sigma_Y/E$ . However, instead of the total strains, the code computes and stores in **XMAT** the slopes  $(d\sigma/d\epsilon)$  of the segments. The resulting organization of **XMAT** is as follows:



- `XMAT(7)` is the number of segments  $N$  in the traction curve
- `XMAT(8)` is the stress value at the end of the first segment, i.e. the initial yield stress  $\sigma_{Y0}$
- `XMAT(9)` is the slope of the first segment, i.e. Young's modulus  $E$  (elastic slope)
- `XMAT(10)` is the stress at the end of the second segment
- `XMAT(11)` is the slope of the second segment, i.e. the first elasto-plastic slope
- ...
- `XMAT(7+2N-1)` is the last stress value, i.e. the stress at the end of the traction curve
- `XMAT(7+2N)` is the slope of the last segment of the curve

If, during the calculation, the total strain value exceeds the last point of the traction curve specified, the code assumes a perfectly plastic behaviour. In other words, the given traction curve is extrapolated by a semi-infinite horizontal (zero slope) final segment.

This is coded as follows:

```

    IACT = 7
    NPT = NINT(XMAT(IACT))
    NN = NPT*2
    ID = IACT + 4
    IF1 = IACT + NN
    FAIL_LIMIT_ELEM = FAIL_LIMIT

```

That is:

- `IACT` is the address in `XMAT` of the quantity  $N$ , which is 7 in this case
- `NPT` is  $N$
- `NN` is the number of values in `XMAT` used to store the traction curve ( $2N$ )
- `ID` is the address in `XMAT` of the second slope (first plastic slope)
- `IF1` is the address in `XMAT` of the last slope

Next, we check that the current yield stress (`ECR(4)`) has a meaningful value (greater than zero). Another routine (`ECROU`) is responsible for initializing this quantity after the material parameters have been read and before starting the transient calculation, so we check this here for safety. An error message (`CALL ERRMSS`) is printed on the listing and the code stops with a message on the console (`STOP 'text'`) in case of error:

```

    IF (ECR(4) == 0.D0) THEN
        CALL ERRMSS ('M_MATERIAL_VM23', 'ECR(4) NOT INITIALIZED')
        STOP 'SGDI : ECR(4) NOT INITIALIZED'
    ENDIF

```

To conclude the initialization phase, we check whether the current Gauss point was previously failed. Note that this may happen, since an element is not necessarily eroded, and therefore skipped, when only one or some of its Gauss points have failed (depending on the user's choice of erosion parameters in the input file).

If the Gauss point was previously failed, we set some values (for safety) and then completely skip the following elasto-plastic calculation:

```

      IF (FAIL_CRIT /= 0) THEN
        PEPSMAX = -HUGE(PEPSMAX)
* check previous failure
      IF (ECR(6) /= 0.D0) THEN
* current gauss point was previously failed, so let's just skip it ...
        SEQ = 0.D0
        PRESS = 0.D0
        FAIL_LIMIT_ELEM = 0.D0
        PEPSMAX = HUGE(PEPSMAX)
        GO TO 990
      ENDIF
    ENDIF
  ENDIF

```

## 2.4 Treatment of plane stress or uniaxial stress conditions

As already mentioned, the plane stress case (IPLANC equal 1) and the uniaxial stress case (IPLANC equal 2) are special.

In the plane stress case  $\sigma_z$  is assumed to be zero (more precisely, it is assumed to be negligible with respect to the other components). The corresponding strain increment  $\Delta\epsilon_z$  cannot be computed from geometrical quantities in the (calling) element routine like the other strain increment components, but it must be evaluated in the present material routine.

The uniaxial stress case is similar but the zero stress condition applies to two stress components ( $\sigma_z$  and  $\sigma_y$ ) and therefore two strain increments ( $\Delta\epsilon_z$  and  $\Delta\epsilon_y$ ) must be evaluated.

This is coded as follows:

```

      GOTO (100, 101, 102, 101) , IPLANC + 1
102 AUX = DEPS(2)                                ! provisional strain incr.
      DEPS(2) = -(SIG(4) + D*DEPS(1)) / (G2 + 2.D0*D) ! actual strain increment
      EPST(2) = EPST(2) - AUX + DEPS(2)            ! actual total strain
101 AUX = DEPS(4)
      DEPS(4) = -(SIG(4) + D*(DEPS(1) + DEPS(2))) / (G2 + D)
      EPST(4) = EPST(4) - AUX + DEPS(4)
100 CONTINUE

```

The meaning of the FORTRAN “computed GO TO” statement is as follows. Compute the value of (IPLANC+1). If this is one, jump to the first label (100), if it is two, jump to the second label (101), and so on.

Thus we see that in the normal (no plane stress) case (IPLANC equal to 0) the whole calculation is skipped (label 100). In the plane stress case (IPLANC equal to 1) the code re-computes  $\Delta\epsilon_z$  (DEPS(4)) and  $\epsilon_z$  (label 101). In the uniaxial stress case (IPLANC equal to 2) the code first re-computes  $\Delta\epsilon_y$  (DEPS(2)) and  $\epsilon_y$  (label 102), and then it re-computes also  $\Delta\epsilon_z$  and  $\epsilon_z$ .

In the plane stress case, the formulas used are as follows:

$$\begin{aligned}
\Delta\epsilon_z^* &= \Delta\epsilon_z \\
\Delta\epsilon_z &= -\frac{\sigma_z + \lambda(\Delta\epsilon_x + \Delta\epsilon_y)}{\lambda + 2\mu} \\
\epsilon_z &= \epsilon_z - \Delta\epsilon_z^* + \Delta\epsilon_z
\end{aligned} \tag{1}$$

The strain increment  $\Delta\epsilon_z$  is calculated by the second expression above as a function of the (old) stress  $\sigma_z$  (which should be zero, or negligible with respect to the other stress components), and of the other strain increments  $\Delta\epsilon_x$  and  $\Delta\epsilon_y$ .

The first and third expressions are used to keep track of the total strain  $\epsilon_z$ , which is used only for post-processing purposes. Recall that total strains in the exchange list are already the new values, i.e. they already include the strain increments for the current step. Therefore, in order to be consistent, we first store the input value of  $\Delta\epsilon_z$  in a local variable  $\Delta\epsilon_z^*$ . This value should be zero, if the element routine does its job properly, and so this operation is probably redundant, but it is safer to do so. Next, we compute the “true”  $\Delta\epsilon_z$ . Finally, we compute the “true” total strain  $\epsilon_z$  by first removing the provisional increment and then adding the true increment to the value received in input from the element routine.

In the uniaxial stress case, the formulas are:

$$\begin{aligned}
\Delta\epsilon_y^* &= \Delta\epsilon_y \\
\Delta\epsilon_y &= -\frac{\sigma_z + \lambda\Delta\epsilon_x}{2\lambda + 2\mu} \\
\epsilon_y &= \epsilon_y - \Delta\epsilon_y^* + \Delta\epsilon_y
\end{aligned} \tag{2}$$

and then

$$\begin{aligned}
\Delta\epsilon_z^* &= \Delta\epsilon_z \\
\Delta\epsilon_z &= -\frac{\sigma_z + \lambda(\Delta\epsilon_x + \Delta\epsilon_y)}{\lambda + 2\mu} \\
\epsilon_z &= \epsilon_z - \Delta\epsilon_z^* + \Delta\epsilon_z
\end{aligned} \tag{3}$$

That is, we first compute the strain increment  $\Delta\epsilon_y$  with (2) as a function of the (old)  $\sigma_z$  (which should be negligible) and of  $\Delta\epsilon_x$ . Then, we compute  $\Delta\epsilon_z$  with (1) or (3) as a function of the old  $\sigma_z$ , of  $\Delta\epsilon_x$  and of the just re-computed  $\Delta\epsilon_y$ .

Having computed the “true” strain increments, the rest of the elasto-plastic calculation proceeds in exactly the same way whether or not a plane or uniaxial stress condition has been specified. The resulting lateral stresses will not be exactly zero, but they should be negligible (many orders of magnitude smaller) compared with the other stress components. Furthermore, it is claimed that this procedure does not propagate these “errors” so that they are not cumulated over the following steps and the nominally zero stresses will always remain negligible.

## 2.5 Elastic predictor

In a first phase, we provisionally assume that the material behaves elastically and we compute a “trial” state of stress  $\boldsymbol{\sigma}^{\text{tr}}$  corresponding to the old stress state  $\boldsymbol{\sigma}$  plus purely elastic stress increments corresponding to the given (or computed, see previous Section) strain increments  $\Delta\boldsymbol{\epsilon}$ :

$$\boldsymbol{\sigma}^{\text{tr}} = \boldsymbol{\sigma} + \boldsymbol{D} : \Delta\boldsymbol{\epsilon} \quad (4)$$

where  $\boldsymbol{D}$  is Hooke's elasticity matrix. This is coded as follows:

```

V = D*(DEPS(1) + DEPS(2) + DEPS(4))
PI(1) = SIG(1) + G2*DEPS(1) + V
PI(2) = SIG(2) + G2*DEPS(2) + V
PI(4) = SIG(4) + G2*DEPS(4) + V
A01 = PI(1) - PI(2)
A02 = PI(2) - PI(4)
A03 = PI(4) - PI(1)
SEQ = (A01*A01 + A02*A02 + A03*A03)*.5D0
GOTO (200, 201, 202, 203) , ITAU + 1
203 PI(6) = SIG(6) + G2*DEPS(6)*.5D0
SEQ = SEQ + 3.D0*PI(6)*PI(6)
202 PI(5) = SIG(5) + G2*DEPS(5)*.5D0
SEQ = SEQ + 3.D0*PI(5)*PI(5)
201 PI(3) = SIG(3) + G2*DEPS(3)*.5D0
SEQ = SEQ + 3.D0*PI(3)*PI(3)
200 SEQ = SQRT(SEQ)
PRESS = (PI(1) + PI(2) + PI(4)) / 3.D0

```

First the “volumetric” term  $V$  is evaluated:

$$V = \lambda(\Delta\epsilon_x + \Delta\epsilon_y + \Delta\epsilon_z) = \lambda\Delta e \quad (5)$$

where  $\Delta e$  indicates the volumetric strain increment  $\Delta e = \Delta\epsilon_x + \Delta\epsilon_y + \Delta\epsilon_z$ .

The trial stresses are provisionally stored in the PI array. The longitudinal components are:

$$\begin{aligned}
\sigma_x^{\text{tr}} &= \sigma_x + \lambda\Delta e + 2\mu\Delta\epsilon_x = \sigma_x + (\lambda + 2\mu)\Delta\epsilon_x + \lambda(\Delta\epsilon_y + \Delta\epsilon_z) \\
\sigma_y^{\text{tr}} &= \sigma_y + \lambda\Delta e + 2\mu\Delta\epsilon_y = \sigma_y + (\lambda + 2\mu)\Delta\epsilon_y + \lambda(\Delta\epsilon_x + \Delta\epsilon_z) \\
\sigma_z^{\text{tr}} &= \sigma_z + \lambda\Delta e + 2\mu\Delta\epsilon_z = \sigma_z + (\lambda + 2\mu)\Delta\epsilon_z + \lambda(\Delta\epsilon_x + \Delta\epsilon_y)
\end{aligned} \quad (6)$$

The square of the (trial) equivalent (or von Mises) stress  $\bar{\sigma}^{\text{tr}}$  (SEQ) is provisionally calculated (in the absence of shear stresses) as:

$$(\bar{\sigma}^{\text{tr}})^2 = \frac{1}{2} \left[ (\sigma_x^{\text{tr}} - \sigma_y^{\text{tr}})^2 + (\sigma_y^{\text{tr}} - \sigma_z^{\text{tr}})^2 + (\sigma_z^{\text{tr}} - \sigma_x^{\text{tr}})^2 \right] \quad (7)$$

Next, we consider any shear stresses that could be present, as indicated by the value of ITAU. For each shear component to be treated, we compute the corresponding (trial, elastic) new stress value and add its contribution to the expression of the equivalent stress. Like in the calculation of plane or uniaxial stress cases presented in the previous Section, use is made of a FORTRAN computed GO TO statement.

For example, in the most general case (ITAU equal 3, i.e. all three shear stress components are present) the procedure is:

$$\begin{aligned}
\tau_{xz}^{\text{tr}} &= \tau_{xz} + \mu\Delta\gamma_{xz} = \tau_{xy} + G\Delta\gamma_{xz} \\
\tau_{yz}^{\text{tr}} &= \tau_{yz} + \mu\Delta\gamma_{yz} = \tau_{yz} + G\Delta\gamma_{yz} \\
\tau_{xy}^{\text{tr}} &= \tau_{xy} + \mu\Delta\gamma_{xy} = \tau_{xy} + G\Delta\gamma_{xy}
\end{aligned} \tag{8}$$

and then

$$(\bar{\sigma}^{\text{tr}})^2 = \frac{1}{2} \left[ (\sigma_x^{\text{tr}} - \sigma_y^{\text{tr}})^2 + (\sigma_y^{\text{tr}} - \sigma_z^{\text{tr}})^2 + (\sigma_z^{\text{tr}} - \sigma_x^{\text{tr}})^2 \right] + 3 (\tau_{xz}^{\text{tr}})^2 + 3 (\tau_{yz}^{\text{tr}})^2 + 3 (\tau_{xy}^{\text{tr}})^2 \tag{9}$$

The case ITAU equal 2 does not occur in practice. In the case ITAU equal 1, the first two eqs. (8) are skipped and the corresponding shear components are not included in (9). Finally, in the case ITAU equal 0, all eqs. (8) and (9) are skipped.

Finally, the trial equivalent stress is evaluated:

$$\bar{\sigma}^{\text{tr}} = \sqrt{(\bar{\sigma}^{\text{tr}})^2} \tag{10}$$

and the trial hydrostatic stress  $p$  (PRESS) is also computed:

$$p = (\sigma_x^{\text{tr}} + \sigma_y^{\text{tr}} + \sigma_z^{\text{tr}}) / 3 \tag{11}$$

Note that this is also the final value since a possible following plastic correction operates only on the deviatoric terms and therefore it does not modify the value of the hydrostatic stress.

At this point the routine checks whether the trial equivalent stress SEQ is above the current yield stress ECR(4) (i.e. whether  $\bar{\sigma}^{\text{tr}} > \sigma_Y$ ):

```

      IF (SEQ > ECR(4)) GO TO 5
*
* elastic predictor (von mises equivalent stress) is below elastic limit
      ECR(2) = SEQ
      GO TO 99
*
      5 CONTINUE
*
* plastic correction

```

If this is the case, the trial stress state lies outside the yield surface, and is therefore inadmissible, so that a plastic correction must be applied (GO TO 5 and see next Section).

Otherwise, the trial stress state is elastic (and therefore admissible). The new stresses and other quantities coincide with the already calculated trial values, so we may skip the plastic correction (GO TO 99) after storing the new equivalent stress in ECR(2) for subsequent use.

## 2.6 Plastic correction by radial return

When the trial (elastic) stress computed in the previous Section lies outside the current yield surface, it is projected onto the surface by the radial return method.

The assumed yield function is von Mises yield function. The material starts yielding when the stress state  $\boldsymbol{\sigma}$  is such that

$$f(\boldsymbol{\sigma}) = k^2 \quad (12)$$

with the function  $f$  given by:

$$f(\boldsymbol{\sigma}) = J_2(\mathbf{s}) = \frac{1}{2} s_{ij} s_{ij} \quad (13)$$

where  $J_2(\mathbf{s})$  is the second invariant of the deviatoric stress tensor  $\mathbf{s}$ , defined as:

$$s_{ij} = \sigma_{ij} - \frac{1}{3} \sigma_{kk} \delta_{ij} \quad (14)$$

By using (13) and (14), eq. (12) becomes

$$\begin{aligned} f(\boldsymbol{\sigma}) &= \frac{1}{6} \left[ (\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 \right] + \tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2 \\ &= \frac{1}{3} (\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x \sigma_y - \sigma_y \sigma_z - \sigma_z \sigma_x) + \tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2 \\ &= k^2 \end{aligned} \quad (15)$$

In the space of principal stresses  $\sigma_1, \sigma_2, \sigma_3$ , the surface  $f$  is represented by a circular cylinder with the axis along the line of equation  $\sigma_1 = \sigma_2 = \sigma_3$  (which tri-sects the first octant of the space and is called the hydrostatic axis) and of radius  $R$ :

$$R = \sqrt{2}k \quad (16)$$

The intersection between the Mises cylinder and a plane  $\pi$  normal to the above mentioned axis (deviatoric plane), of equation:

$$\sigma_1 + \sigma_2 + \sigma_3 = c \quad (17)$$

is a circle of radius  $R$ .

In order to perform the plastic correction, the current tangent modulus  $E_t$  of the traction curve, i.e. the slope of the curve, must be determined. This is coded as follows:

```

CURR = ECR(4)
IF (ID > IF1) GO TO 8
DO 6 I = ID, IF1, 2
IF (CURR > XMAT(I-1)) GO TO 6
ET = XMAT(I)
GO TO 7
6 CONTINUE
*
* attention ! the traction curve is extrapolated using a zero slope !
8 ET = 0.
!
! compute EP=dsig/dplas using ET=dsig/deps and YMX (Young's modulus)
7 CONTINUE
EP = YMX*ET / (YMX - ET)
COEF = EP / (G3 + EP)

```

The value of the current yield stress  $\sigma_Y$  (ECR(4)) is stored in a local variable CURR. Then, if ID is greater than IF1, i.e. if the user has specified only one point on the traction curve (the elastic limit), so that the material is assumed to perform as perfectly plastic (zero slope) beyond this point, we set  $E_t = 0$ .

Otherwise, we loop (DO 6) over the points defining the traction curve, from the second one onwards, and stop as soon as the point's stress is greater than the current yield stress. The slope of the segment which ends in this point is taken as the current tangent slope  $E_t$ . If the test is not satisfied, not even for the last point defined, this means that the current stress state exceeds the traction curve defined by the user. Again, the curve is extrapolated by a perfectly plastic behaviour (zero slope) and we set  $E_t = 0$ .

Next, we compute the plastic modulus  $E_p$  (EP), which represents the variation of equivalent stress  $\bar{\sigma}$  per unit increment of equivalent plastic strain  $\bar{\epsilon}^p$ , and is given by:

$$E_p = \frac{d\bar{\sigma}}{d\bar{\epsilon}^p} = \frac{EE_t}{E - E_t} \quad (18)$$

and a coefficient  $\chi$  (COEF) that will be used below to compute the hardening.

$$\chi = \frac{E_p}{E_p + 3\mu} \quad (19)$$

The current yield stress  $\sigma_Y$  (ECR(4)) is now updated by taking into effect the hardening of the material.

```

IF (LSGDIO == 0) THEN
  ECR(4) = (SEQ - ECR(4))*COEF + ECR(4)
ELSE
* 'old' material model used in eurdyn (slightly less precise)
* (to be used only for precise comparison with old eurdyn runs)
  ECR(4) = (SEQ - ECR(4))*(ET/YMX) + ECR(4)
ENDIF

```

The new yield stress is normally given by:

$$\sigma_Y = \sigma_Y + \chi (\bar{\sigma} - \sigma_Y) = \sigma_Y + \frac{E_p}{E_p + 3\mu} (\bar{\sigma} - \sigma_Y) \quad (20)$$

By activating a special input option (OPTI OLDS) it is possible to use an alternative expression for the new yield stress, which was adopted in the EURDYN codes (precursors of EPX):

$$\sigma_Y = \sigma_Y + \frac{E_t}{E} (\bar{\sigma} - \sigma_Y) \quad (21)$$

The expression (21) is slightly less accurate than (20) and so the option should only be activated for comparison with old results or for debugging purposes.

In the present case of von Mises yield surface, the projection of trial stress onto the (updated, due to hardening) yield surface reduces to radial scaling of the stress in the deviatoric plane. This is coded as follows:

```

IF (IPLANC /= 3) THEN ! except the plane stress case for CEA's shells
  C = ECR(4) / SEQ
  PI(1) = C*(PI(1) - PRESS) + PRESS

```

```

        PI(2) = C*(PI(2) - PRESS) + PRESS
        PI(4) = C*(PI(4) - PRESS) + PRESS
        GOTO (300, 301, 302, 303) , ITAU + 1
303    PI(6) = C*PI(6)
302    PI(5) = C*PI(5)
301    PI(3) = C*PI(3)
300    ECR(2) = ECR(4)
* compute the equivalent plastic strain
    DBETA = (SEQ - CURR) / (G3 + EP)
    ECR(3) = ECR(3) + DBETA
    ELSE ! plane stress case for CEA's shells
        . . . (see next Section)
    ENDIF

```

As anticipated, a special treatment is done for CEA's 3D shells for which IPLANC is set to the special value 3 (see ELSE clause in the above code fragment). This is described in the next Section.

In the normal case (IPLANC equal 0, 1 or 2), the following procedure is adopted. The scaling factor  $c$  for the stress is computed according to:

$$c = \frac{\sigma_Y}{\bar{\sigma}^{\text{tr}}} \quad (22)$$

i.e., it is the ratio between the (current) yield stress  $\sigma_Y$  and the trial (elastic) equivalent stress  $\bar{\sigma}^{\text{tr}}$ .

For the longitudinal stress components ( $\sigma_x$ ,  $\sigma_y$  and  $\sigma_z$ ) the scaling is applied only to the deviatoric part of the stress, since the hydrostatic part is unaffected:

$$\begin{aligned} \sigma_x &= c(\sigma_x^{\text{tr}} - p) + p \\ \sigma_y &= c(\sigma_y^{\text{tr}} - p) + p \\ \sigma_z &= c(\sigma_z^{\text{tr}} - p) + p \end{aligned} \quad (23)$$

Here  $p$  is the trial hydrostatic stress defined by eq. (11). Note that the scaling is applied to all components, irrespective of the value of IPLANC.

Then, the shear components are scaled as well:

$$\begin{aligned} \tau_{xy} &= c\tau_{xy}^{\text{tr}} \\ \tau_{yz} &= c\tau_{yz}^{\text{tr}} \\ \tau_{xz} &= c\tau_{xz}^{\text{tr}} \end{aligned} \quad (24)$$

As usual, a computed GO TO on ITAU is used in order to apply the scaling only to the shear components which are effectively present.

Next, the von Mises equivalent stress  $\bar{\sigma}$  (ECR(2)) is updated to the current yield stress value  $\sigma_Y$  (ECR(4)):

$$\bar{\sigma} = \sigma_Y \quad (25)$$

Finally, the increment of equivalent plastic strain  $\Delta\bar{\epsilon}^p$  (DBETA) is computed according to:

$$\Delta\bar{\epsilon}^p = \frac{\bar{\sigma}^{\text{tr}} - \sigma_Y}{3\mu + E_p} \quad (26)$$



and the total equivalent plastic strain  $\bar{\epsilon}^p$  (ECR(3)) is incremented:

$$\bar{\epsilon}^p = \bar{\epsilon}^p + \Delta\bar{\epsilon}^p \quad (27)$$

## 2.7 Plane stress case for CEA's 3D shells

In the case of CEA's 3D shells (IPLANC equal 3, i.e. elements DKT3, DST3, Q4GS, T3GS, Q4GR and QPPS) the plane stress condition, i.e. the fact that the stress through the element thickness is vanishing, is enforced "exactly", with a special treatment, as anticipated in the previous Section.

```

IF (IPLANC /= 3) THEN ! except the plane stress case for CEA's shells
  . . . (see previous Section)
ELSE ! plane stress case for CEA's shells
  CALL HILL_CP_JRC (ITAU, XNU, YMX, PI, CURR, EP, DBETA)
  ECR(3) = ECR(3) + DBETA
  ECR(4) = CURR + EP*DBETA
  ECR(2) = ECR(4)
ENDIF

```

In this case, a separate routine HILL\_CP\_JRC is called, which performs the following operations. The routine receives the number of shear stress components ITAU (which for these shell elements can be either 1 or 3), the elastic constants  $\nu$  (XNU) and  $E$  (YMX), the trial (elastic) stresses  $\sigma^{\text{tr}}$  (PI), the current yield stress  $\sigma_Y$  (CURR) and the plastic modulus  $E_p$  (EP).

The routine computes and returns the new elasto-plastic stresses  $\sigma$  (in the PI array, which is modified) and the increment of equivalent plastic strain  $\Delta\bar{\epsilon}^p$  (DBETA). A detailed description of the HILL\_CP\_JRC routine is out of scope here. Basically, an iterative treatment is performed which should ensure a vanishing value of the stress through the thickness in a certain number of iterations (maximum 10 iterations are allowed). Before returning, the routine explicitly sets  $\sigma_z = 0$  (in PI(4)).

The total equivalent plastic strain  $\bar{\epsilon}^p$  (ECR(3)) is incremented:

$$\bar{\epsilon}^p = \bar{\epsilon}^p + \Delta\bar{\epsilon}^p \quad (28)$$

The current yield stress  $\sigma_Y$  (CURR) is incremented and stored in ECR(4) according to:

$$\sigma_Y = \sigma_Y + E_p \Delta\bar{\epsilon}^p \quad (29)$$

and finally the von Mises equivalent stress  $\bar{\sigma}$  (ECR(2)) is updated to the current yield stress value  $\sigma_Y$  (ECR(4)):

$$\bar{\sigma} = \sigma_Y \quad (30)$$

## 2.8 Finalizations

Before returning to the calling program, the routine performs some finalizations.

```

      SEQ = ECR(4) ! new Von Mises stress (for failure/erosion test)
      99 ECR(1) = PRESS
*
* sound speed

```

```

      CSON = CS
      GOTO (500, 501, 502, 501) , IPLANC + 1
500 CSON = CSON*T3*T3 / T2
501 CSON = CSON / (T1*T3)
502 CSON = SQRT(CSON)

```

The value of the new equivalent (von Mises) stress `ECR(4)` is stored in the local variable `SEQ`. Then, the hydrostatic stress  $p$  as computed by (11) is stored in `ECR(1)` for post-processing purposes.

Then, the sound speed  $c_s$  (`CSON`) in the material is evaluated by a computed `GO TO` depending on `IPLANC`:

- In the general (no plane stress condition) case (`IPLANC` equals 0)

$$c_s = \sqrt{\frac{E(1-\nu)}{\rho(1+\nu)(1-2\nu)}} \quad (31)$$

- In the plane stress case (`IPLANC` equals 1 or 3)

$$c_s = \sqrt{\frac{E}{\rho(1-\nu^2)}} \quad (32)$$

- In the uniaxial stress case (`IPLANC` equals 2)

$$c_s = \sqrt{\frac{E}{\rho}} \quad (33)$$

Next, failure of the current Gauss point is checked if needed:

```

      SELECT CASE (FAIL_CRIT)
      . . . (code skipped for brevity)
      END SELECT
990 IF ((FAIL_CRIT /= 0) .AND. (ECR(6) /= 0.D0)) THEN
      . . . (code skipped for brevity)
      ENDIF

```

and erosion of the current element is also checked, if this mechanism has been activated by the user:

```

      IF (L_EROSION) THEN
      . . . (code skipped for brevity)
      ENDIF

```

Details of both these procedures are skipped here because the failure and erosion mechanisms are being completely revised in EPX and are undergoing substantial re-arrangements at the time of this writing.

The routine concludes by storing in `ECR(5)`, for post-processing purposes, the computed speed of sound in the material:

```

      ECR(5) = CSON
*
      END SUBROUTINE SGDI

```

## 2.9 Organization of stress components

As seen in the previous Sections, the SGDI routine assumes a certain organization of the stress components (**SIG** array) which, in the most general case (3D continuum), is:

$$\boldsymbol{\sigma} = \{\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, \tau_{yz}, \tau_{xz}\} \quad (34)$$

This is the most general case, where all six components of stress are present and must be treated. In the 2D continuum case, only the first four components of stress are present:

$$\boldsymbol{\sigma} = \{\sigma_x, \sigma_y, \tau_{xy}, \sigma_z\} \quad (35)$$

It should be noted that all other cases assume either six components like in (34) or four components like in (35) to be present in the **SIG** (and in the **PI**) array, and this irrespective of the value of **IPLANC**. This may look strange at first sight: for example, in a 2D bar element such as **FUN2**, characterized by **ITAU** equal 0 and **IPLANC** equal 2 (see Table 2), there is in principle only one ‘interesting’ stress component,  $\sigma_x$  (**SIG**(1)), so that the other three components,  $\sigma_y$ ,  $\tau_{xy}$  and  $\sigma_z$  seem redundant. However, their presence in the arrays and in the routine is justified for the following reasons:

- First, **EPX** uses a fixed number of stress (**SIG**) components for each element type, irrespective of the loading case (e.g. plane stress, plane strain or axisymmetric).
- Second, by convention and for simplicity the (total) strain array **EPST** and the strain increment array **DEPS** have the same length and the same components organization as the stress array.
- Third, although the shear stress  $\tau_{xy}$  and the corresponding shear strain  $\gamma_{xy}$  are strictly zero for this element, and therefore redundant (in fact, the routine neither reads (uses) nor writes (modifies) **SIG**(3) in this case), the two lateral stresses ( $\sigma_y$  and  $\sigma_z$ ) are only approximately zero (that is, negligible with respect to the other stress components) in the approach used by the routine to implement the plane stress condition(s). Therefore, they must be stored in the **SIG** array so that their (small) values can be re-used in the next time step in order to ensure an (approximate) plane stress condition, without growth of the numerical error introduced. Furthermore, the corresponding total lateral strains ( $\epsilon_y$  and  $\epsilon_z$ ) and lateral strain increments ( $\Delta\epsilon_y$  and  $\Delta\epsilon_z$ ) are certainly not negligible, so that the corresponding slots must be present in the **EPST** and **DEPS** arrays, and therefore also in **SIG** due to the previous point. The lateral (total) strains are needed in the element routine in order to take into account the lateral “thinning” (or thickening, in case of compression) of the element when computing the internal nodal forces equivalent to the state of stress in the deformed bar element. The same holds for shell elements: while the stress through the thickness is at least approximately zero, the corresponding total strain is not zero and must be returned to the element routine in order to take into account the variation of element thickness when computing the internal forces.

The element types which can use the **VM23** material and the associated number of stresses and list of stress components are summarized in Table 4 for 2D elements and in Table 5 for 3D elements. The stress components are listed in the order in which they are stored in the element’s (global) **SIG** array. Note that this may or may not coincide with the order expected by the SGDI material routine. In the latter case, the element routine (not the material routine) is responsible for building up a local stress array (as well as local arrays for total strain and for strain increments) with the suitable organization of components, to be passed to the material routine.

In Table 4, the abbreviation ‘axis.’ stays for (2D) axisymmetric calculation. In the axisymmetric case the ‘out of plane’ stress  $\sigma_z$  is sometimes indicated as  $\sigma_\theta$  (hoop stress). Those (longitudinal) stress components which are nominally zero, and in practice negligible (orders of magnitude smaller) with respect to the other stress components, are highlighted in the Tables. For example,  $\sigma_z \approx 0$  indicates a plane stress (IPLANC equal 1) condition.

### 2.9.1 The case of CEA’s 3D shells

In Table 5 it should be noted that 3D shell elements from CEA (DKT3, DST3, Q4GS, T3GS, Q4GR, QPPS) use a variable organization of the stress components (and of the related quantities such as total strains etc.), depending upon the type of formulation (‘case’) assumed: either ‘linear’ or ‘nonlinear’. In practice, the case depends on the material used by the element. At the moment of this writing:

- The LINE, MCOU, CHAN and GLRC materials are associated with the linear case. In this case, the element is not integrated through the thickness in order to obtain the equivalent nodal forces. The membrane stress contributions ( $\sigma_x^m, \sigma_y^m, \tau_{xy}^m$ ), which do not depend upon the distance from the element’s reference (or mid-) plane, are separated from the bending stress contributions ( $\sigma_x^b, \sigma_y^b, \tau_{xy}^b$ ), which depend (linearly) upon that distance. Note that, for these elements, the two transverse shear stress components (when present, i.e. when a “thick” or Mindlin shell element formulation is adopted) are stored in the inverse order with respect to the other 3D elements, that is ( $\tau_{xz}, \tau_{yz}$ ) instead of ( $\tau_{yz}, \tau_{xz}$ ). The total number of stress components for “thick” shells is then eight, not six. Note also that in the linear case the stress through the thickness  $\sigma_z$ , which is assumed to be zero or negligible, is not stored in the SIG array. Consequently, the total strain through the thickness  $\epsilon_z$  (which, however, is not negligible in general) is not available in the EPST array. This means that element “thinning” in the thickness direction cannot be accounted for in the linear case.
- All other materials (including the VM23 material considered here) are associated with the nonlinear case. In this case, the element is integrated through the thickness (to track plasticity) in order to obtain the equivalent nodal forces. No separation between membrane and bending terms is done in this case and each Gauss point through the thickness is treated separately, with its own time history of the constitutive law. This means that the first three components of stress are the total values ( $\sigma_x, \sigma_y, \tau_{xy}$ ), and not the membrane values. Consequently, the second three components (slots 4 to 6) of stress become free and can be used for other purposes. Component four of the stress is used to store the stress through the thickness  $\sigma_z$  (which is zero or vanishing), so that the corresponding value in the total strain table is the strain through the thickness  $\epsilon_z$ . Therefore, in the case of nonlinear material, element “thinning” in the thickness direction can be dealt with, by making appropriate use of  $\epsilon_z$  in the element routine.

Due to the difference in treatment between the linear and nonlinear cases, it is not advisable to mix up ‘linear’ and ‘nonlinear’ materials in the same CEA’s 3D shell element, if the element has layers of different materials (sandwich model). The results in such a case would be unpredictable.

Finally, it should be noted that the two reduced-integrated CEA shells (Q4GR and QPPS) have two additional stress (or rather pseudo-stress) components in the SIG table (and therefore also in EPST), at slots 9 and 10. These are the so-called anti-hourglass stresses. The total number of stresses is therefore ten for these elements.

The case of the 3D shell element T3MC (also from CEA) is special but is not described in detail.

Element	ITAU	IPLANC	Loading case	N. of stresses	Stress components
FUN2	0	2	Any	4	$\sigma_x, \sigma_y \approx 0, \tau_{xy} = 0, \sigma_z \approx 0$
ED01	0	1	Plane strain or axis.	4	$\sigma_x, (\sigma_y \text{ or } \sigma_\theta), \tau_{xy} = 0, \sigma_z \approx 0$
	0	2	Plane (here uniaxial) stress	4	$\sigma_x, \sigma_y \approx 0, \tau_{xy} = 0, \sigma_z \approx 0$
ED41	0	1	Plane strain or axis.	4	$\sigma_x, (\sigma_y \text{ or } \sigma_\theta), \tau_{xy} = 0, \sigma_z \approx 0$
	0	2	Plane (here uniaxial) stress	4	$\sigma_x, \sigma_y \approx 0, \tau_{xy} = 0, \sigma_z \approx 0$
Q41	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q42	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q41N	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q42N	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q41L	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q42L	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q42G	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q92	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q93	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
Q92A	1	0	Axisymmetric	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_\theta$
Q95	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
CAR1	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
CAR4	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
TRIA	1	0	Plane strain or axis.	4	$\sigma_x, \sigma_y, \tau_{xy}, (\sigma_z \text{ or } \sigma_\theta)$
	1	1	Plane stress	4	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$

Table 4: Stress organization for 2D elements which can use the VM23 material

## 2.10 Numerical tests

### 2.10.1 Uniaxial large strain tests

We consider the test problem depicted in Figure 1. A tri-unit cube is kept fixed in the  $x$ -direction along its “left” face, while the “right” face is submitted to an imposed velocity in the  $x$  direction, constant in time and equal to 1 m/s, until it reaches a length of 2.0 units (i.e., the final time of the test is set to 1.0 s). All motions in the  $y$  and  $z$  directions are unrestrained.

Two material types are considered. The first material is elastic, with density  $\rho = 8000 \text{ kg/m}^3$ , Young’s modulus  $E = 2.0 \times 10^{11} \text{ Pa}$  and Poisson’s coefficient  $\nu = 0.3$ . The second material has the same elastic properties as the first one but is elastic perfectly plastic with a yield stress  $\sigma_Y = 4.0 \times 10^8 \text{ Pa}$ , which corresponds to a yield strain of 0.2 %. The deformed cube is shown (qualitatively) in Figure 2.

The longitudinal (natural) strain at the final elongation  $\Delta x = 1.0$  is

$$\epsilon_x = \ln \left( \frac{L}{L_0} \right) = \ln \frac{2.0}{1.0} = 0.69315 \quad (36)$$

An approximate analytical solution of the problem is obtained, as a reference. The approxi-

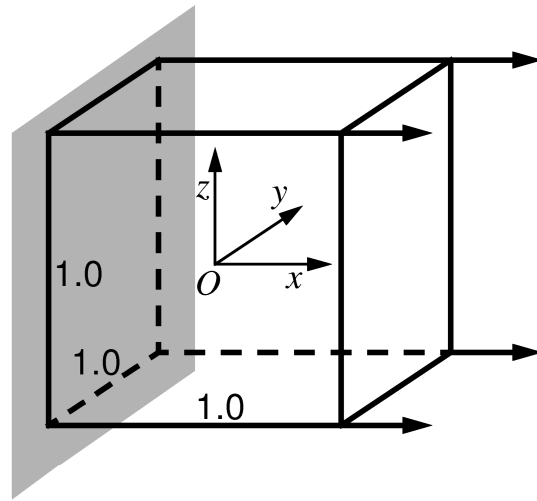


Figure 1: Uniaxial stress problem.

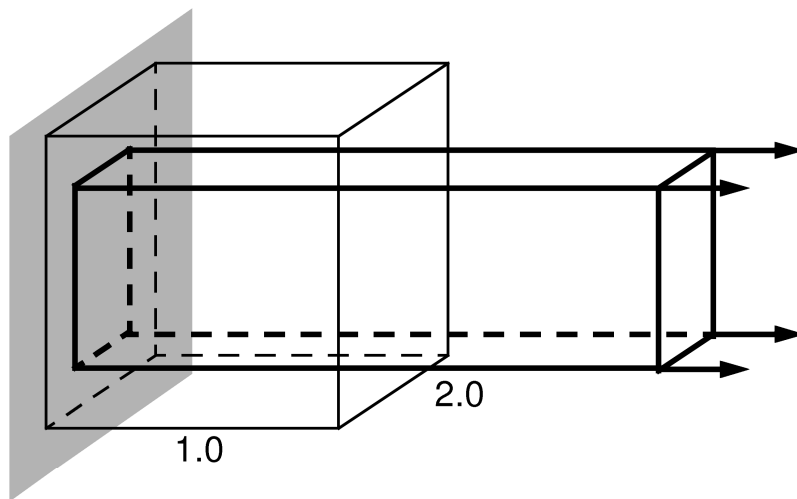


Figure 2: Deformed cube.

Element	ITAU	IPLANC	N. of stresses	Case	Stress components
FUN3	0	2	4	Any	$\sigma_x, \sigma_y \approx 0, \tau_{xy} = 0, \sigma_z \approx 0$
COQI	1	1	4	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0$
DKT3	1	3	6	Linear	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b$
				Nonlinear	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, 0, 0$
C272	3	0	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, \tau_{yz}, \tau_{xz}$
C273	3	0	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, \tau_{yz}, \tau_{xz}$
C81L	3	0	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, \tau_{yz}, \tau_{xz}$
C82L	3	0	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, \tau_{yz}, \tau_{xz}$
CUB6	3	0	6	Any	$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}$
CUB8	3	0	6	Any	$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}$
CUBE	3	0	6	Any	$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}$
PR6	3	0	6	Any	$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}$
PRIS	3	0	6	Any	$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}$
TETR	3	0	6	Any	$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}$
CQD4	3	1	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0, \tau_{yz}, \tau_{xz}$
CQD9	3	1	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0, \tau_{yz}, \tau_{xz}$
CQD3	3	1	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0, \tau_{yz}, \tau_{xz}$
CQD6	3	1	6	Any	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z \approx 0, \tau_{yz}, \tau_{xz}$
T3MC	3	1	8	Any	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b, \tau_{xz}, \tau_{yz}$
DST3	3	3	8	Linear	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b, \tau_{xz}, \tau_{yz}$
				Nonlinear	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, 0, 0, \tau_{xz}, \tau_{yz}$
Q4GS	3	3	8	Linear	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b, \tau_{xz}, \tau_{yz}$
				Nonlinear	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, 0, 0, \tau_{xz}, \tau_{yz}$
T3GS	3	3	8	Linear	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b, \tau_{xz}, \tau_{yz}$
				Nonlinear	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, 0, 0, \tau_{xz}, \tau_{yz}$
Q4GR	3	3	10	Linear	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b, \tau_{xz}, \tau_{yz}, \sigma_h^1, \sigma_h^2$
QPPS	3	3	10	Nonlinear	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, 0, 0, \tau_{xz}, \tau_{yz}, \sigma_h^1, \sigma_h^2$
				Linear	$\sigma_x^m, \sigma_y^m, \tau_{xy}^m, \sigma_x^b, \sigma_y^b, \tau_{xy}^b, \tau_{xz}, \tau_{yz}, \sigma_h^1, \sigma_h^2$
				Nonlinear	$\sigma_x, \sigma_y, \tau_{xy}, \sigma_z, 0, 0, \tau_{xz}, \tau_{yz}, \sigma_h^1, \sigma_h^2$

Table 5: Stress organization for 3D elements which can use the VM23 material

mation lies in the fact that dynamic effects (inertia) are ignored in the analytical (static) solution. Inertia is of course included in the numerical simulation and produces lateral oscillations, as the cube deforms longitudinally. However, since the imposed velocity is relatively small (quasi-static), these oscillations are negligible.

#### *Elastic case*

If the material is elastic, the lateral strains are given by

$$\epsilon_y = \epsilon_z = -\nu\epsilon_x = -0.3 \times 0.69315 = -0.20794 \quad (37)$$

while all shear strains are null, of course

$$\gamma_{xy} = \gamma_{yz} = \gamma_{xz} = 0.0 \quad (38)$$

The longitudinal stress is

$$\sigma_x = E\epsilon_x = 2.0 \times 10^{11} \cdot 0.69315 = 1.3863 \times 10^{11} \quad (39)$$

and all other stress components are null

$$\sigma_y = \sigma_z = \tau_{xy} = \tau_{yz} = \tau_{xz} = 0.0 \quad (40)$$

In order to compute the longitudinal force necessary to stretch the cube, it is necessary to evaluate the current cross section. The lateral lengths of the deformed cube are obtained from the lateral (natural) strains (37)

$$L_y = L_0 \exp(\epsilon_y) = L_z = L_0 \exp(\epsilon_z) = 1.0 \cdot \exp(-0.20794) = 0.81225 \quad (41)$$

so that the area is

$$A = L_y L_z = 0.81225^2 = 0.65975 \quad (42)$$

The longitudinal force at the final time is then

$$F_x = A \sigma_x = 0.65975 \cdot 1.3863 \times 10^{11} = 9.1461 \times 10^{10} \quad (43)$$

Of course, the force applied at each node of the right face of the cube is 1/4 of the (total) value given by (43).

#### *Elastic perfectly plastic case*

If the material is elastic perfectly plastic, with the material parameters listed above, one may assume that when an elongation of 1.0 is reached (which corresponds to an engineering strain of 100 %) the elastic components of strains are negligible compared with the plastic components. This means that the volume  $V$  of the deformed specimen can be considered (approximately) constant and equal to the initial volume  $V_0 = 1 \text{ m}^3$ .

The lateral strains are in this case:

$$\epsilon_y = \epsilon_z = -0.5\epsilon_x = -0.5 \times 0.69315 = -0.34657 \quad (44)$$

i.e., it is like if the material would be elastic with a Poisson's coefficient  $\nu = 0.5$ .

The lateral lengths of the deformed specimen become

$$L_y = L_0 \exp(\epsilon_y) = L_z = L_0 \exp(\epsilon_z) = 1.0 \cdot \exp(-0.34657) = 0.70711 \quad (45)$$

so that the area is

$$A = L_y L_z = 0.70711^2 = 0.50000 \quad (46)$$

This obvious result could have been obtained directly from the assumption that the volume stays constant: if the length doubles, then the cross section must be halved.

Since the material has no hardening, the longitudinal stress is equal to the yield stress

$$\sigma_x = \sigma_Y = 4.0 \times 10^8 \quad (47)$$

and the (total) longitudinal force at the final time is

$$F_x = A \sigma_x = 0.50000 \cdot 4.0000 \times 10^8 = 2.0000 \times 10^8 \quad (48)$$

#### *Elastic numerical solutions*

The numerical solutions obtained with the elastic material model are listed in Table 6.

The first solution, case **CUBE04**, uses CEA's hexahedron element CUBE and the VM23 material. The JAUM option is specified in order to activate large-strain treatment for these elements. The time increment is fixed to  $\Delta t = 20 \text{ } \mu\text{s}$ , so that 50,000 steps are needed to reach the final time.



Case	Element	Material	Comments	$\epsilon_x$	$\epsilon_y$	$\epsilon_z$	$\sigma_x$	$F_x$
Analytical	N/A	N/A	Static solution	0.69315	-0.20794	-0.20794	$1.3863 \times 10^{11}$	$9.1461 \times 10^{10}$
CUBE04	CUBE	VM23 (elastic)	Option JAUM	0.69314	-0.20796	-0.20796	$1.3862 \times 10^{11}$	$9.1454 \times 10^{10}$
C81L04	C81L	VM23 (elastic)		0.69315	-0.20797	-0.20797	$1.3862 \times 10^{11}$	$9.1454 \times 10^{10}$
<b>CAR104</b>	CAR1	VM23 (elastic)	Option JAUM	0.69314	-0.20791	-0.20795	$1.3863 \times 10^{11}$	$11.261 \times 10^{10}$
Q41L04	Q41L	VM23 (elastic)		0.69315	-0.20793	-0.20795	$1.3863 \times 10^{11}$	$9.1463 \times 10^{10}$
CQD4S1	CQD4	VM23 (elastic)		0.69315	-0.20793	-0.20795	$1.3863 \times 10^{11}$	$9.1462 \times 10^{10}$
<b>Q4GSS1</b>	Q4GS	VM23 (elastic)		0.69314	-0.20791	-0.20795	$1.3863 \times 10^{11}$	$11.261 \times 10^{10}$
<b>Q4GSS3</b>	Q4GS	LINE	'Linear' case	0.69314	-0.20791	N/A	$1.3863 \times 10^{11}$	$11.261 \times 10^{10}$
<b>Q4GSS4</b>	Q4GS	VMIS PARF (elastic)		0.69314	-0.20791	$4.4 \times 10^{-15}$	$1.3863 \times 10^{11}$	$11.261 \times 10^{10}$
FUVM07	FUN3	VM23 (elastic)		0.69315	-0.20795	-0.20795	$1.3863 \times 10^{11}$	$9.1462 \times 10^{10}$
FUVM08	FUN2	VM23 (elastic)		0.69315	-0.20795	-0.20795	$1.3863 \times 10^{11}$	$9.1462 \times 10^{10}$

Table 6: Solutions of the uniaxial stress problem with elastic material

The imposed time step is well below the estimated stability limit which in this case is  $119 \mu\text{s}$  in the initial configuration and  $97 \mu\text{s}$  in the final configuration. This is necessary in order to avoid numerical instability of the explicit solution which might occur since the lateral motions are left free.

As shown in Table 6, the numerical results are in very good agreement with the reference solution. Figure 3 shows the variation in time of strains, stresses and nodal forces and the resulting longitudinal stress *vs.* longitudinal strain diagram. Because of large deformations, the strain, stress and force curves are not linear in time, although the imposed longitudinal displacement is linear in time. The Cauchy stress *vs.* natural strain diagram is linear, because it has been so chosen in the input file.

The next solution, case **C81L04**, is similar to CUBE04 but uses JRC's hexahedron element C81L. Since all JRC's elements are large strain by default, no specific option (JAUM) is necessary. Results are in excellent agreement with the analytical solution and with the previous numerical solution, as it can be seen from Table 6.

The two solutions presented so far used 3D continuum elements. Now we consider solutions in 2D of the same problem by using the plane stress formulation (calculation of type CPLA), in which the assumption of zero (or vanishing) out-of-plane stress is embedded in the model.

The first such solution is case **CAR104**, which uses CEA's quadrilateral element CAR1 (like usual with CEA's continuum elements, the JAUM option is required to treat large strains).

As shown in Table 6, the solution is in good agreement with the reference as concerns the strains and the stresses. However, the computed longitudinal force is very different:  $11.261 \times 10^{10}$  N instead of  $9.1454 \times 10^{10}$  N.

The discrepancy, which is quite large in this case, seems to come from the fact the the used element (CAR1) does not take into account element "thinning" (i.e. the value of  $\epsilon_z$ ) when integrating the stresses (which are correct) over the element's volume in order to obtain the equivalent nodal forces. This can be checked very simply, because if one does not account for  $\epsilon_z$  (thus assuming in practice that  $\epsilon_z = 0$ , and therefore  $L_z = L_{z0} = 1.0$ ), then the deformed element's cross section is

$$A = L_y L_z = 0.81225 \cdot 1.0 = 0.81225 \quad (49)$$

and the longitudinal force becomes

$$F_x = A \sigma_x = 0.81225 \cdot 1.3863 \times 10^{11} = 1.1260 \times 10^{11} \quad (50)$$

which is indeed the value found by the code.

The next solution, case **Q41L04**, is similar to the previous one (2D CPLA calculation) but uses JRC's element Q41L instead of CEA's element CAR1 (the JAUM option is not necessary). The solution is in excellent agreement with the reference (see Table 6), thus showing that the Q41L

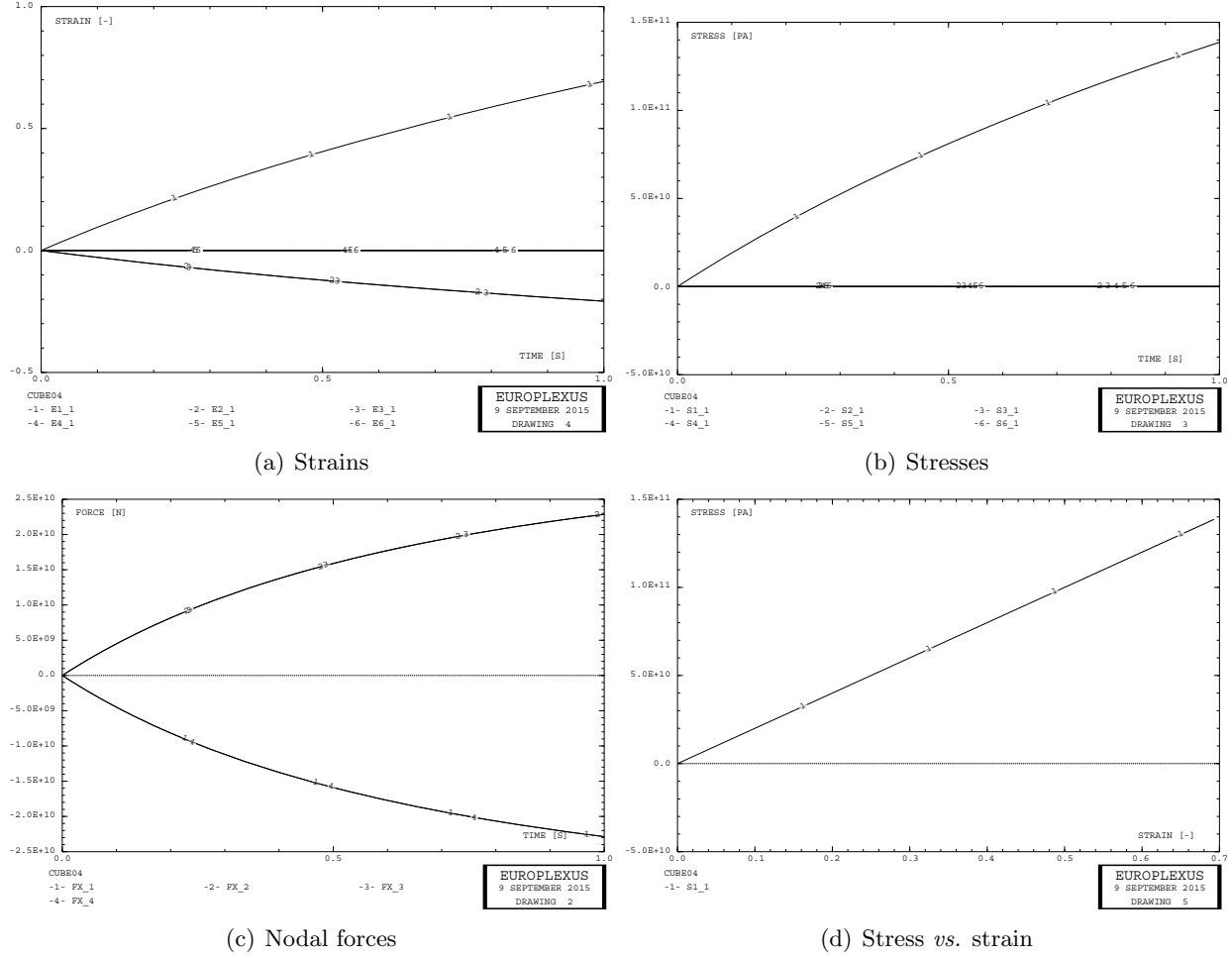


Figure 3: Some results of test case CUBE04.

element (unlike CAR1) does take into account correctly the variation of element thickness due to large deformation in the element's plane.

The same problem considered so far can also be tentatively solved by using a 3D shell element, although of course a shell with the shape of a unit cube (thickness aspect ratio equal to 1.0) is not to be recommended in real applications.

The next solution, case **CQD4S1**, uses JRC's shell element CQD4, which is a 4-node quadrilateral shell degenerated from continuum (Hughes-Liu formulation). The solution, listed in Table 6, is in excellent agreement with the reference, showing in particular that this element takes into account out-of-plane thinning.

Then, case **Q4GSS1** uses CEA's quadrilateral shell Q4GS. As it appears from Table 6, the results are quite similar to those obtained with CAR1: all results are correct, except the longitudinal force which is largely overestimated (the same value as in case CAR104 is obtained), due to the fact that the element thinning is not taken into account when computing the equivalent internal forces.

The faulty behaviour of CAR1, Q4GS and other CEA's elements of the same families will have to be corrected by intervening in the respective element routines.

Finally, two more solutions are obtained in order to verify that the faulty behaviour observed is

not due to the use of the VM23 material. The first case, **Q4GSS3**, uses the LINE material, which in conjunction with the Q4GS element, qualifies the case as ‘linear’, as discussed in a previous Section. Recall that in such a case, as indicated in Table 6, the strain through the thickness ( $\epsilon_z$ ) is not available and the first three stress components are the membrane contributions (however, in this example they are the total stresses because there is no bending in this problem). It is therefore not surprising that the same results as in cases CAR104 and Q4GSS1 are obtained, in particular the same (overestimated) value of longitudinal force.

The final solution, case **Q4GSS4**, uses CEA’s VMIS PARF (elastic perfectly plastic) material, but with the properties set in such a way that the stress remains in the elastic regime during the whole transient solution. In this case, since the material is (at least nominally) elasto-plastic, the element is integrated through the thickness. However, as shown in Table 6, the material routine does not seem to compute the transverse normal strain  $\epsilon_z$  since the returned value is practically zero. Then, the same (overestimated) force value is obtained as in the previous faulty cases.

To conclude the elastic solutions, we obtain results in 3D with the cable (or bar) element FUN3 (case **FUVM07**) and in 2D with the element FUN2 (case **FUVM08**), by using the VM23 material. In both cases the results are in excellent agreement with the analytical solution, as shown in Table 6, including the value of the nodal forces. This shows that such elements take into due account the variation of the element cross section.

#### *Elastic perfectly plastic numerical solutions*

The numerical solutions obtained with the elastic perfectly plastic material model are listed in Table 7.

Case	Element	Material	Comments	$\epsilon_x$	$\epsilon_y$	$\epsilon_z$	$\sigma_x$	$F_x$
Analytical	N/A	N/A	Static solution	0.69315	-0.34657	-0.34657	$4.0000 \times 10^8$	$2.0000 \times 10^8$
CUBE05	CUBE	VM23	Option JAUM	0.69314	-0.34617	-0.34617	$3.9857 \times 10^8$	$1.9945 \times 10^8$
C81L05	C81L	VM23		0.69315	-0.34618	-0.34618	$3.9901 \times 10^8$	$1.9966 \times 10^8$
<b>CAR105</b>	CAR1	VM23	Option JAUM	0.69314	-0.34760	-0.34473	$4.0038 \times 10^8$	<b><math>2.8282 \times 10^8</math></b>
Q41L05	Q41L	VM23		0.69315	-0.34761	-0.34474	$4.0038 \times 10^8$	$2.0035 \times 10^8$
CQD4S2	CQD4	VM23		0.69315	-0.34761	-0.34474	$4.0038 \times 10^8$	$2.0035 \times 10^8$
<b>Q4GSS2</b>	Q4GS	VM23		0.69314	-0.34699	-0.14835	$4.0000 \times 10^8$	<b><math>2.8273 \times 10^8</math></b>
<b>Q4GSS5</b>	Q4GS	VMIS PARF		0.69314	-0.34699	$-4.1 \times 10^{-15}$	$4.0000 \times 10^8$	<b><math>2.8273 \times 10^8</math></b>
FUVM09	FUN3	VM23		0.69315	-0.34617	-0.34617	$4.0077 \times 10^8$	$2.0054 \times 10^8$
FUVM10	FUN2	VM23		0.69315	-0.34617	-0.34617	$4.0077 \times 10^8$	$2.0054 \times 10^8$

Table 7: Solutions of the uniaxial stress problem with elastic perfectly plastic material

The first solution, case **CUBE05**, is the elasto-plastic analogous of the elastic case CUBE04 presented previously. As shown in Table 7, the numerical results are in very good agreement with the reference solution. Figure 4 shows the variation in time of strains, stresses and nodal forces and the resulting longitudinal stress *vs.* longitudinal strain diagram. The numerical solution exhibits some oscillations which, however, remain relatively negligible.

The next solution, case **C81L05**, uses the C81L element from JRC. As shown in Table 7, the numerical results are in very good agreement with the reference solution.

Then comes solution **CAR105** which uses the 2D quadrilateral CAR1 in plane stress (CPLA) conditions. As can be seen from Table 7, the numerical results are in good agreement with the reference, with the notable exception of the longitudinal force which, like in the elastic case CAR104 seen previously, is largely overestimated:  $2.8282 \times 10^8$  instead of  $2.0000 \times 10^8$ .

Again, this is due to the neglect of the thickness variation in the element. In fact, by neglecting the  $z$ -term, the cross section of the deformed element becomes in this case

$$A = L_y L_z = 0.70711 \cdot 1.0 = 0.70711 \quad (51)$$

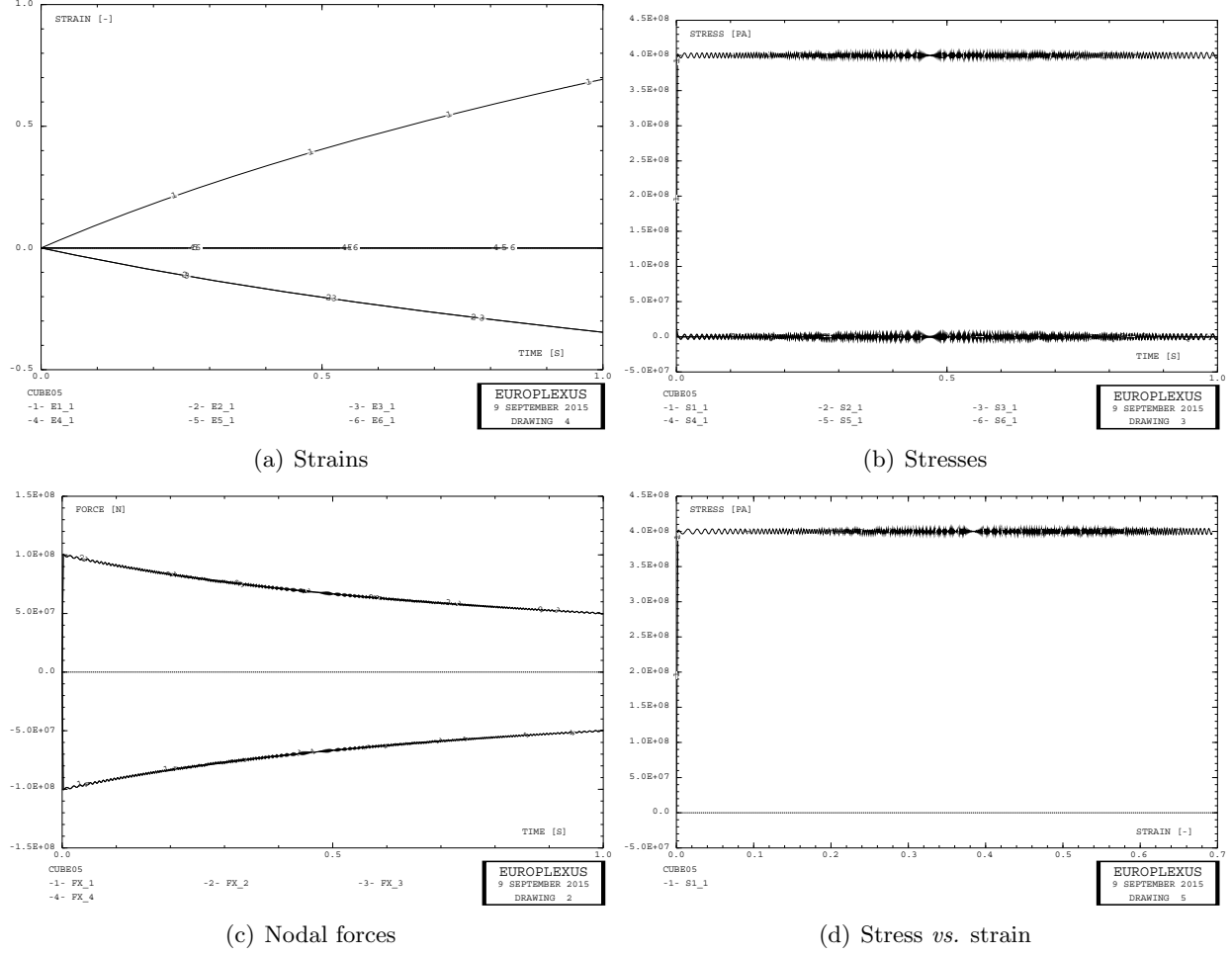


Figure 4: Some results of test case CUBE05.

and the longitudinal force

$$F_x = A\sigma_x = 0.70711 \cdot 4.0 \times 10^8 = 2.8284 \times 10^8 \quad (52)$$

which is the value found by the code.

The cases **Q41L05** and **CQD4S2**, which use JRC's Q41L quadrilateral and CQD4 degenerated shell, respectively, are in good agreement with the reference, see Table 7.

Then, the case **Q4GSS2** uses CEA's Q4GS shell element. As shown in Table 7, two of the results are incorrect in this case. First, the strain through the thickness  $\sigma_z$  is 0.14835 instead of 0.34657 and this looks strange because this quantity is computed correctly in all other cases. Second, the longitudinal force is wrong (as it could be expected). The value is practically the same as in case CAR105 and shows (once more) that the element does not take into account transversal thinning. However, in order to consider the thinning, the transversal strain should be computed correctly in the first place.

Finally, one more solution is obtained in order to verify that the faulty behaviour observed is not due to the use of the VM23 material. The case **Q4GSS5** uses CEA's VMIS PARF (elastic perfectly plastic) material, so that the Q4GS element is integrated through the thickness. However, as shown in Table 7, the material routine does not seem to compute the transverse normal strain  $\epsilon_z$

since the returned value is practically zero. Then, the same (overestimated) force value is obtained as in the previous faulty cases.

To conclude the elastic perfectly plastic solutions, we obtain results in 3D with the cable (or bar) element FUN3 (case **FUVM09**) and in 2D with the element FUN2 (case **FUVM10**), by using the VM23 material. In both cases the results are in excellent agreement with the analytical solution, as shown in Table 7, including the value of the nodal forces. This shows that such elements take into due account the variation of the element cross section.

#### *Corrections in CEA's elements in 2D*

The tests presented above have shown that some elements from CEA (e.g. the continuum element CAR1) do not take into account the variation of element thickness when computing the equivalent nodal forces. This may lead to inaccuracy if large membrane strains occur in the element.

In the CAR1 element routine, an extra multiplicative factor  $T$  (variable's name **THICK**) has been tentatively added in the calculation of nodal internal forces, which is defined as follows:

$$T = \begin{cases} 1.0 & \text{in Plane Strain or Axisymmetric} \\ \exp(\epsilon_z) & \text{in Plane Stress} \end{cases} \quad (53)$$

The same correction has been applied also to the CAR4 (fully-integrated quadrilateral) and TRIA (triangle) elements. With such corrections, the results obtained in the previously problematic test cases become in good agreement with the reference solution, see Table 8 for the elastic case and Table 9 for the elastoplastic case.

Case	Element	Material	Comments	$\epsilon_x$	$\epsilon_y$	$\epsilon_z$	$\sigma_x$	$F_x$
Analytical	N/A	N/A	Static solution	0.69315	-0.20794	-0.20794	$1.3863 \times 10^{11}$	$9.1461 \times 10^{10}$
CAR104	CAR1	VM23 (elastic)	Option JAUM	0.69314	-0.20793	-0.20795	$1.3863 \times 10^{11}$	$9.1462 \times 10^{10}$

Table 8: Corrected solutions of the uniaxial stress problem with elastic material

Case	Element	Material	Comments	$\epsilon_x$	$\epsilon_y$	$\epsilon_z$	$\sigma_x$	$F_x$
Analytical	N/A	N/A	Static solution	0.69315	-0.34657	-0.34657	$4.0000 \times 10^8$	$2.0000 \times 10^8$
CAR105	CAR1	VM23	Option JAUM	0.69314	-0.34760	-0.34473	$4.0038 \times 10^8$	$2.0035 \times 10^8$

Table 9: Corrected solutions of the uniaxial stress problem with elastic perfectly plastic material

#### *Corrections in CEA's shell elements in 3D*

An attempt has been made to include a similar correction to the one proposed for 2D continuum elements, also in CEA's 3D shell elements, in order to take into account element thickness variation at least for those material models which compute and return to the element routine the strain through the element thickness.

While the correction was successful for the present test case of uniaxial strain of the element, it produced unwanted changes also in more physical test cases, involving a combination of membrane and bending loading in the shell. For this reason, the correction has not been implemented yet and CEA is being contacted in order to solve the problem.

## References

- [1] EUROPLEXUS User's Manual, on-line version: <http://europlexus.jrc.ec.europa.eu>.

# Appendix A

All the input files used in the previous Sections are listed below.

## c81l04.epx

```
C81L04
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 C81L 1 TERM
-0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
-0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 -0.5 0.5 0.5
1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 2.E11
TRAC 1 2.E11 1.0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 5 8 TERM
VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRD FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
```

## c81l05.epx

```
C81L05
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 C81L 1 TERM
-0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
-0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 -0.5 0.5 0.5
1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 4.E8
TRAC 2 4.E8 2.E-3
4.E8 1.E0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 5 8 TERM
VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
```

```
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRD FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
```

## car104.epx

```
CAR104
ECHO
!CONV WIN
LAGR CPLA
GEOM LIBR POIN 4 CAR1 1 TERM
-0.5 -0.5 0.5 -0.5 0.5 0.5 -0.5 0.5
1 2 3 4
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 2.E11
TRAC 1 2.E11 1.0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRD FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
JAUM
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
```

```

COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
CAR104
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 C81L 1 TERM
-0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
-0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 0.5 -0.5 0.5 0.5
1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 2.E11
TRAC 1 2.E11 1.0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 5 8 TERM
VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## car105.epx

```

CAR105
ECHO
!CONV WIN
LAGR CPLA
GEOM LIBR POIN 4 CAR1 1 TERM
-0.5 -0.5 0.5 -0.5 0.5 0.5 -0.5 0.5
1 2 3 4
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 4.E8
TRAC 2 4.E8 2.E-3
4.E8 1.E0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM

```

```

INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
JAUM
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
CAR105
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 C81L 1 TERM
-0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
-0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 0.5 -0.5 0.5 0.5
1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 2.E11
TRAC 1 2.E11 1.0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 5 8 TERM
VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER

```

```
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
```

## cqd4s1.epx

```
CQD4S1
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 CQD4 1 TERM
  0 0 0 1 0 0 1 1 0 0 1 0
  1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000 YOUN 2.E11 NU 0.3 ELAS 2.E11
  TRAC 1 2.E11 1.0
  LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
  VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
  1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
  FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
```

## cqd4s2.epx

```
CQD4S2
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 CQD4 1 TERM
  0 0 0 1 0 0 1 1 0 0 1 0
  1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000 YOUN 2.E11 NU 0.3 ELAS 4.E8
  TRAC 2 4.E8 2.E-3
  4.E8 1.E0
  LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
  VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
  1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
  FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
```

```
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
```

## cube04.epx

```
CUBE04
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 CUBE 1 TERM
  -0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
  -0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 0.5 -0.5 0.5 0.5
  1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
  YOUN 2.E11
  NU 0.3
  ELAS 2.E11
  TRAC 1 2.E11 1.0
  LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
  2.0 1.0
LINK COUP
  BLOQ 1 LECT 1 4 5 8 TERM
  VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
  FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
  JAUM
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
```

## cube05.epx



```

CUBE05
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 CUBE 1 TERM
-0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
-0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 0.5 -0.5 0.5 0.5
1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
      YOUN 2.E11
      NU 0.3
      ELAS 4.E8
      TRAC 2 4.E8 2.E-3
      4.E8 1.E0
      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 4 5 8 TERM
      VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
      JAUM
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## fuvm07.epx

```

FUVM07
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 2 FUN3 1 TERM
-0.5 0 0 0.5 0 0
1 2
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000.
      YOUN 2.E11
      NU 0.3
      ELAS 2.E11
      TRAC 1 2.E11 1.0
      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 TERM
      VITE 1 1.0 FONC 1 LECT 2 TERM
INIT VITE 1 1.0 LECT 2 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP

```

```

AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 4 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 32 'vmis_1' ECRO COMP 2 ELEM LECT 1 TERM
COUR 33 'epla_1' ECRO COMP 3 ELEM LECT 1 TERM
TRAC 1 2 AXES 1.0 'DISPL. [M]' YZER
TRAC 3 4 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
TRAC 32 AXES 1.0 'VMIS [PA]' XAXE 33 1.0 'EPLA [-]' YZER
FIN

```

## fuvm08.epx

```

FUVM08
ECHO
!CONV WIN
CPLA LAGR
GEOM LIBR POIN 2 FUN2 1 TERM
-0.5 0 0.5 0
1 2
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000.
      YOUN 2.E11
      NU 0.3
      ELAS 2.E11
      TRAC 1 2.E11 1.0
      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 TERM
      VITE 1 1.0 FONC 1 LECT 2 TERM
INIT VITE 1 1.0 LECT 2 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 4 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 32 'vmis_1' ECRO COMP 2 ELEM LECT 1 TERM
COUR 33 'epla_1' ECRO COMP 3 ELEM LECT 1 TERM
TRAC 1 2 AXES 1.0 'DISPL. [M]' YZER
TRAC 3 4 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
TRAC 32 AXES 1.0 'VMIS [PA]' XAXE 33 1.0 'EPLA [-]' YZER
FIN

```

## fuvm09.epx

```

FUVM09
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 2 FUN3 1 TERM
-0.5 0 0 0.5 0 0
1 2
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000 YOUN 2.E11 NU 0.3 ELAS 4.E8
      TRAC 2 4.E8 2.E-3
      4.E8 1.E0

```

```

      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 TERM
      VITE 1 1.0 FONC 1 LECT 2 TERM
INIT VITE 1 1.0 LECT 2 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECR0 FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 4 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 32 'vmis_1' ECR0 COMP 2 ELEM LECT 1 TERM
COUR 33 'epla_1' ECR0 COMP 3 ELEM LECT 1 TERM
TRAC 1 2 AXES 1.0 'DISPL. [M]' YZER
TRAC 3 4 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
TRAC 32 AXES 1.0 'VMIS [PA]' XAXE 33 1.0 'EPLA [-]' YZER
FIN

```

## fuvm10.epx

```

FUVM10
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 2 FUN3 1 TERM
      -0.5 0 0 0.5 0 0
      1 2
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000 YOUN 2.E11 NU 0.3 ELAS 4.E8
      TRAC 2 4.E8 2.E-3
      4.E8 1.E0
      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 TERM
      VITE 1 1.0 FONC 1 LECT 2 TERM
INIT VITE 1 1.0 LECT 2 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECR0 FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 4 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 32 'vmis_1' ECR0 COMP 2 ELEM LECT 1 TERM
COUR 33 'epla_1' ECR0 COMP 3 ELEM LECT 1 TERM
TRAC 1 2 AXES 1.0 'DISPL. [M]' YZER
TRAC 3 4 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
TRAC 32 AXES 1.0 'VMIS [PA]' XAXE 33 1.0 'EPLA [-]' YZER
FIN

```

## q41l04.epx

```

Q41L04
ECHO
!CONV WIN
LAGR CPLA
GEOM LIBR POIN 4 Q41L 1 TERM
      -0.5 -0.5 0.5 -0.5 0.5 0.5 -0.5 0.5
      1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000.
      YOUN 2.E11
      NU 0.3
      ELAS 2.E11
      TRAC 1 2.E11 1.0
      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 4 TERM
      VITE 1 1.0 FONC 1 LECT 2 3 TERM
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECR0 FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## q41l05.epx

```

Q41L05
ECHO
!CONV WIN
LAGR CPLA
GEOM LIBR POIN 4 Q41L 1 TERM
      -0.5 -0.5 0.5 -0.5 0.5 0.5 -0.5 0.5
      1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000.
      YOUN 2.E11
      NU 0.3
      ELAS 4.E8
      TRAC 2 4.E8 2.E-3
      4.E8 1.E0
      LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
      2.0 1.0
LINK COUP
      BLOQ 1 LECT 1 4 TERM
      VITE 1 1.0 FONC 1 LECT 2 3 TERM
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECR0 FREQ 5000
      FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO

```

```

RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN
Q41L05
ECHO
!CONV WIN
LAGR TRID
GEOM LIBR POIN 8 C81L 1 TERM
-0.5 -0.5 -0.5 0.5 -0.5 -0.5 0.5 0.5 -0.5 -0.5 0.5 -0.5
-0.5 -0.5 0.5 0.5 -0.5 0.5 0.5 0.5 0.5 -0.5 0.5 0.5
1 2 3 4 5 6 7 8
MATE VM23 RO 8000.
YOUN 2.E11
NU 0.3
ELAS 2.E11
TRAC 1 2.E11 1.0
LECT 1 TERM
FONC 1 TABL 2 0.0 1.0 ! Constant function in time
2.0 1.0
LINK COUP
BLOQ 1 LECT 1 4 5 8 TERM
VITE 1 1.0 FONC 1 LECT 2 3 6 7 TERM
INIT VITE 1 1.0 LECT 2 3 6 7 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL LOG 1
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## q4gss1.epx

```

Q4GSS1
ECHO

```

```

!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 Q4GS 1 TERM
0 0 0 1 0 0 1 1 0 0 1 0
1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000 YOUN 2.E11 NU 0.3 ELAS 2.E11
TRAC 1 2.E11 1.0
LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## q4gss2.epx

```

Q4GSS2
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 Q4GS 1 TERM
0 0 0 1 0 0 1 1 0 0 1 0
1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VM23 RO 8000 YOUN 2.E11 NU 0.3 ELAS 4.E8
TRAC 2 4.E8 2.E-3
4.E8 1.E0
LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM

```

```

COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## q4gss3.epx

```

Q4GSS3
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 Q4GS 1 TERM
0 0 0 1 0 0 1 1 0 0 1 0
1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE LINE RO 8000 YOUN 2.E11 NU 0.3
LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## q4gss4.epx

```

Q4GSS4
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 Q4GS 1 TERM
0 0 0 1 0 0 1 1 0 0 1 0
1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VMIS PARF RO 8000 YOUN 2.E11 NU 0.3 ELAS 2.E11
LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM

```

```

ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

## q4gss5.epx

```

Q4GSS5
ECHO
!CONV WIN
TRID LAGR
GEOM LIBR POIN 4 Q4GS 1 TERM
0 0 0 1 0 0 1 1 0 0 1 0
1 2 3 4
COMP EPAI 1.0 LECT 1 TERM
MATE VMIS PARF RO 8000 YOUN 2.E11 NU 0.3 ELAS 4.E8
LECT 1 TERM
LINK COUP BLOQ 1 LECT 1 4 TERM
VITE 1 1.0 FONC 1 LECT 2 3 TERM
FONC NUM 1 TABL 2 0.0 1.0
1.0 1.0
INIT VITE 1 1.0 LECT 2 3 TERM
ECRI COOR DEPL VITE ACCE FINT FEXT FLIA FDEC CONT EPST ECRO FREQ 5000
FICH ALIC FREQ 50
OPTI NOTE PAS UTIL CSTA 0.5EO
CALC TINI 0 TFIN 1.0 PASF 2.E-5 NMAX 50000
SUIT
Post-treatment
ECHO
RESU ALIC GARD PSCR
SORT GRAP
AXTE 1.0 'Time [s]'
COUR 1 'dx_1' DEPL COMP 1 NOEU LECT 1 TERM
COUR 2 'dx_2' DEPL COMP 1 NOEU LECT 2 TERM
COUR 3 'dx_3' DEPL COMP 1 NOEU LECT 3 TERM
COUR 4 'dx_4' DEPL COMP 1 NOEU LECT 4 TERM
COUR 5 'fx_1' FLIA COMP 1 NOEU LECT 1 TERM
COUR 6 'fx_2' FLIA COMP 1 NOEU LECT 2 TERM
COUR 7 'fx_3' FLIA COMP 1 NOEU LECT 3 TERM
COUR 8 'fx_4' FLIA COMP 1 NOEU LECT 4 TERM
COUR 11 's1_1' CONT COMP 1 ELEM LECT 1 TERM
COUR 12 's2_1' CONT COMP 2 ELEM LECT 1 TERM
COUR 13 's3_1' CONT COMP 3 ELEM LECT 1 TERM
COUR 14 's4_1' CONT COMP 4 ELEM LECT 1 TERM
COUR 15 's5_1' CONT COMP 5 ELEM LECT 1 TERM
COUR 16 's6_1' CONT COMP 6 ELEM LECT 1 TERM
COUR 21 'e1_1' EPST COMP 1 ELEM LECT 1 TERM
COUR 22 'e2_1' EPST COMP 2 ELEM LECT 1 TERM
COUR 23 'e3_1' EPST COMP 3 ELEM LECT 1 TERM
COUR 24 'e4_1' EPST COMP 4 ELEM LECT 1 TERM
COUR 25 'e5_1' EPST COMP 5 ELEM LECT 1 TERM
COUR 26 'e6_1' EPST COMP 6 ELEM LECT 1 TERM
TRAC 1 2 3 4 AXES 1.0 'DISPL. [M]' YZER
TRAC 5 6 7 8 AXES 1.0 'FORCE [N]' YZER
TRAC 11 12 13 14 15 16 AXES 1.0 'STRESS [PA]' YZER
TRAC 21 22 23 24 25 26 AXES 1.0 'STRAIN [-]' YZER
TRAC 11 AXES 1.0 'STRESS [PA]' XAXE 21 1.0 'STRAIN [-]' YZER
FIN

```

Europe Direct is a service to help you find answers to your questions about the European Union

Freephone number (\*): 00 800 6 7 8 9 10 11

(\*) Certain mobile telephone operators do not allow access to 00 800 numbers or these calls may be billed.

A great deal of additional information on the European Union is available on the Internet.

It can be accessed through the Europa server <http://europa.eu/>.

#### **How to obtain EU publications**

Our priced publications are available from EU Bookshop (<http://bookshop.europa.eu>),

where you can place an order with the sales agent of your choice.

The Publications Office has a worldwide network of sales agents.

You can obtain their contact details by sending a fax to (352) 29 29-42758.

European Commission

**EUR 27434 EN – Joint Research Centre – Institute for the Protection and Security of the Citizen**

Title: Description of the elasto-plastic material routine SGDI

Author(s): Folco Casadei, Vegard Aune, Georgios Valsamos, Martin Larcher

Luxembourg: Publications Office of the European Union

2015 – 34 pp. – 21.0 x 29.7 cm

EUR – Scientific and Technical Research series – ISSN 1831-9424 (online)

ISBN 978-92-79-51226-1 (PDF)

doi:10.2788/403240

#### **Abstract**

The report describes the FORTRAN subroutine SGDI for rate-independent deviatoric plasticity with von Mises yield criterion and isotropic hardening, which corresponds to the VM23 material type in the EUROPLEXUS code (abbreviated as EPX in the following). EPX is a Finite Element code for the numerical simulation of fast transient dynamic events in fluid-structural system, jointly developed by the French Commissariat à l'Energie Atomique et aux Energies Alternatives (CEA Saclay) and by the Joint Research Centre of the European Commission (JRC Ispra).

The subroutine I part of the FORTRAN90 module M\_MATERIAL\_VM23. In order to simplify the description of the routine and of the material model, the parts dealing with Gauss point failure and element erosion have been neglected.

### JRC Mission

As the Commission's in-house science service, the Joint Research Centre's mission is to provide EU policies with independent, evidence-based scientific and technical support throughout the whole policy cycle.

Working in close cooperation with policy Directorates-General, the JRC addresses key societal challenges while stimulating innovation through developing new methods, tools and standards, and sharing its know-how with the Member States, the scientific community and international partners.

*Serving society  
Stimulating innovation  
Supporting legislation*